

Purdue University
Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

2012

A Simplified CFD Model for Simulation of the Suction Process Of Reciprocating Compressors

Evandro L. L. Pereira
evandro_l_lange@embraco.com.br

Claudio J. Santos

Cesar J. Deschamps

Rodrigo Kremer

Follow this and additional works at: <http://docs.lib.purdue.edu/icec>

Pereira, Evandro L. L.; Santos, Claudio J.; Deschamps, Cesar J.; and Kremer, Rodrigo, "A Simplified CFD Model for Simulation of the Suction Process Of Reciprocating Compressors" (2012). *International Compressor Engineering Conference*. Paper 2086.
<http://docs.lib.purdue.edu/icec/2086>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

A Simplified Computational Fluid Dynamics Model for the Suction Process of Reciprocating Compressors

Evandro L.L. PEREIRA^{1*}, Claudio J. SANTOS², Cesar J. DESCHAMPS², Rodrigo KREMER¹

¹Embraco
Rua Rui Barbosa, 1020
89219-901, Joinville, SC, Brazil
Evandro_L_Lange@embraco.com.br

²POLO Research Laboratories for Emerging Technologies in Cooling and Thermophysics
Federal University of Santa Catarina
88040-900, Florianópolis, SC, Brazil
deschamps@polo.ufsc.br

* Corresponding Author

ABSTRACT

The suction process in reciprocating compressors is strongly affected by the valve dynamics and the pulsating flow throughout the suction muffler. This paper describes a simplified computational fluid dynamics (CFD) model to simulate the flow through the muffler, suction valve and a small region inside the cylinder. The proposed method is applied to predict the suction process of a small reciprocating compressor and its adequacy is compared with a more time elaborate model in terms of accuracy, computation cost and usability.

1. INTRODUCTION

Several models have been put forward in the literature to predict the performance of reciprocating compressors. The first proposals consisted of quite simple mathematical models based on lumped formulations for the compression cycle in the cylinder and semi-empirical relations to characterize the mass flow rate through the valve system and the force on the valve. Pressure pulsation in the suction and discharge systems affects the compressor performance and can be estimated via several methods, including acoustic theory (Elson and Soedel, 1974) and non-linear partial differential equations for unsteady one-dimensional compressible flow (McLaren *et al.*, 1975; Pérez-Segarra *et al.*, 1994; Bassi *et al.*, 2000).

Numerical models based on two- and three-dimensional formulations were initially available only for detached compressor components, such as mufflers (Choi *et al.*, 2000; Fagotti and Possamai, 2000) and valves (Matos *et al.*, 2002). However, three-dimensional numerical simulations for the whole compressor have recently emerged (Suh *et al.*, 2006; Pereira *et al.*, 2008a; Shiomi *et al.*, 2009; Almbauer *et al.*, 2010), due to the increasingly availability of computational resources and the development of commercial codes that make it easy to integrate CFD and CAD models. Despite the recent advances in numerical simulation methodologies, the computational cost of a full three-dimensional simulation for the compressor is still impracticable for optimization in which several design alternatives have to be analyzed.

Suction mufflers have a direct impact on the volumetric and isentropic efficiencies of reciprocating compressors and are the subject of frequent optimizations. A common method of predicting the fluid flow in the suction muffler is to take into account its actual three-dimensional geometry and prescribe the mass flow rate through the valve obtained via a lumped formulation for the compression chamber. Details of such simulation models can be found in Pereira *et al.* (2008b) and Lang *et al.* (2008). A more comprehensive and computationally expensive model includes the

compression cylinder in the computational domain and considers the coupling between the valve dynamics and fluid flow, giving rise to difficulties in the discretization of the solution domain. Pereira *et al.* (2007), Shiomi *et al.* (2009) and Almbauer *et al.* (2010) provide a detailed description of this modeling approach.

This paper presents a simplified computational fluid dynamics (CFD) model to simulate the suction process of reciprocating compressors that is proposed as a compromise between the two aforementioned simulation approaches in terms of accuracy, computational cost and usability.

2. MATHEMATICAL MODEL

Figure 1 shows the solution domains of the suction system considered in the complete model and in the simplified model. As one can notice, the complete model solves the flow inside the cylinder in the presence of the piston kinematics (Figure 1a), whereas the simplified model includes only a small region of the cylinder located between the suction valve and its seat (Figure 1b). In the present study, both models were developed with a commercial CFD code in which the conservation equations of mass, momentum and energy are solved via the finite volume method.

Valve dynamics is modeled by a one-degree of freedom mass-spring model:

$$m_{eq}\ddot{x} + c\dot{x} + kx = F_p + F_o \quad (1)$$

where m_{eq} , c and k are the valve equivalent mass, damping coefficient and stiffness, respectively, which have to be specified. On the other hand, F_p is the flow induced force on the valve and F_o can accommodate any other force, such as reed pre-tension and also stiction that may occur due to the presence of a lubricating oil film between the valve and the valve seat. Finally, x , \dot{x} and \ddot{x} are the instantaneous valve lift, velocity and acceleration, respectively. The equivalent mass m_{eq} is determined from data of valve stiffness, k , and natural frequency, f_n . The differential equation for the valve dynamics, Eq. (1), was solved by using an explicit Euler method with the force F_p kept constant during each time step.

As one can see in Figure 1b, the simplified model does not include cylinder in the solution domain. Instead, the compression processes is simulated through a lumped model. Hence, although fluid flow through the suction valve is solved via a three-dimensional formulation, fluid properties inside the cylinder are assumed to be spatially homogeneous throughout in each time step of the compression cycle in accordance with the adopted lumped formulation.

The flow induced force acting on the valve, F_p , is evaluated from

$$F_p = \int_A p \, dA \quad (2)$$

The pressure acting on the valve surface facing the piston is considered to be equal to the pressure in the cylinder predicted by the lumped model. On the other hand, the pressure distribution, p , on the valve surface facing the valve seat is predicted with the three-dimensional model.

The compressible turbulent flow that prevails in the suction system was solved through the concept of Reynolds-averaged quantities, in which the value of a computed variable represents an ensemble average over many cycles at a specified spatial location. The turbulence transport contribution was modeled through the RNG k- ϵ model, which has been extensively used and validated for flow through compressor valves (Matos *et al.*, 2002). An ideal gas equation of state completes the system required to solve the compressible flow.

The evaporating pressure is imposed as boundary condition at the inlet of the suction muffler. Turbulence intensity of 3% and turbulence length scale equal to the hydraulic diameter were adopted as the inlet boundary condition for the turbulent flow. All velocity components were set to zero at the solid walls, but for the valve surface the velocity was obtained from Equation (1). A pressure boundary condition was applied at the suction valve outlet of the simplified CFD model, as depicted in Figure 2. The pressure value was obtained from the solution of the lumped model for the cylinder. The evaporating pressure and the suction line temperature were employed as initial estimates

for the gas properties inside the suction muffler. For the compression chamber the condensing pressure and the discharge temperature were set as the initial conditions.

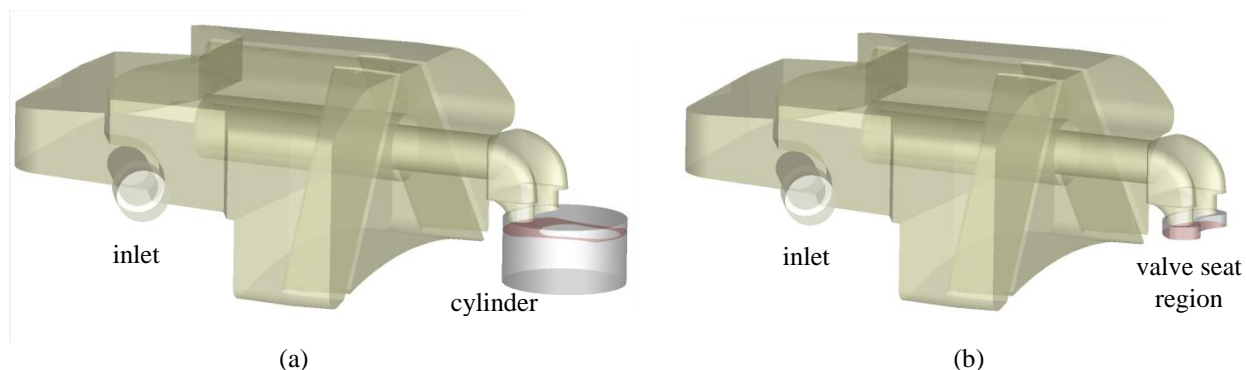


Figure 1: Solution domain for the complete CFD model (a) and the simplified CFD model (b).

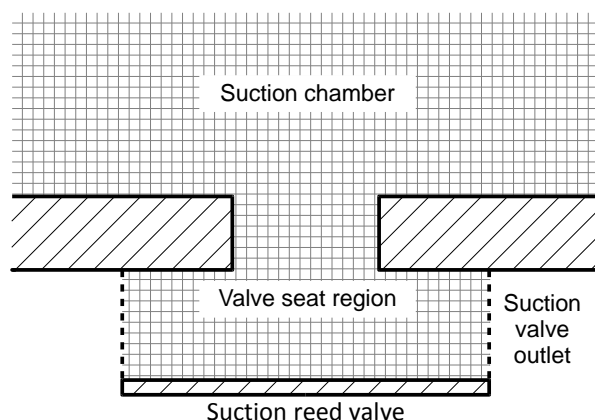


Figure 2: In-cylinder flow region between the suction valve and its seat solved by the simplified model.

A second-order upwind scheme was adopted to interpolate the flow quantities needed at the control volume faces. The coupling between the pressure and velocity fields in the solution procedure was achieved with the SIMPLEC scheme. The system of algebraic equations was solved with a segregated implicit algorithm. The simulation procedure employed time steps of 0.5 degree of crankshaft angle when the valves were closed and 0.1 otherwise.

The solution of the flow in the suction muffler is coupled with the thermodynamic process inside the cylinder which is solved with the lumped model. In other words, the pressure predicted by the lumped model in the cylinder is used at each time step as the boundary condition for the suction valve outlet, unless the valve is closed. The solution of flow field in the suction muffler in turn is used to evaluate the mass flow rate required for the lumped solution procedure of the cylinder domain. The compression process is then calculated for the next time step and the resulting pressure in the cylinder allows a new estimate of pressure to be used as the boundary condition for the suction muffler. This procedure is repeated for a number of compression cycles until a fully periodic cyclic condition is found.

3. RESULTS AND DISCUSSIONS

In order to analyze the simplified model in terms of accuracy and computational cost, the suction process of a small reciprocating compressor was simulated. For the purpose of comparison with the complete 3D model, only a part of the first cycle was considered, starting from the top dead center until the beginning of the discharge process. For the specific geometry and mesh adopted in the present analysis, a reduction of 25% in the computational processing

time was observed when the simplified CFD model was adopted, corresponding to approximately one and an half hour on a computer with an Intel Core i7 950 @ 3.07 GHz processor.

Figure 3 shows predictions for valve displacement, mass flow rate, pressure in the suction chamber and pressure in the cylinder as a function of the crankshaft angle obtained with the complete and the simplified CFD models. Solid and dashed lines refer to the complete model and the simplified model, respectively. Such results were normalized by the maximum valve displacement, the average mass flow rate and the evaporating pressure.

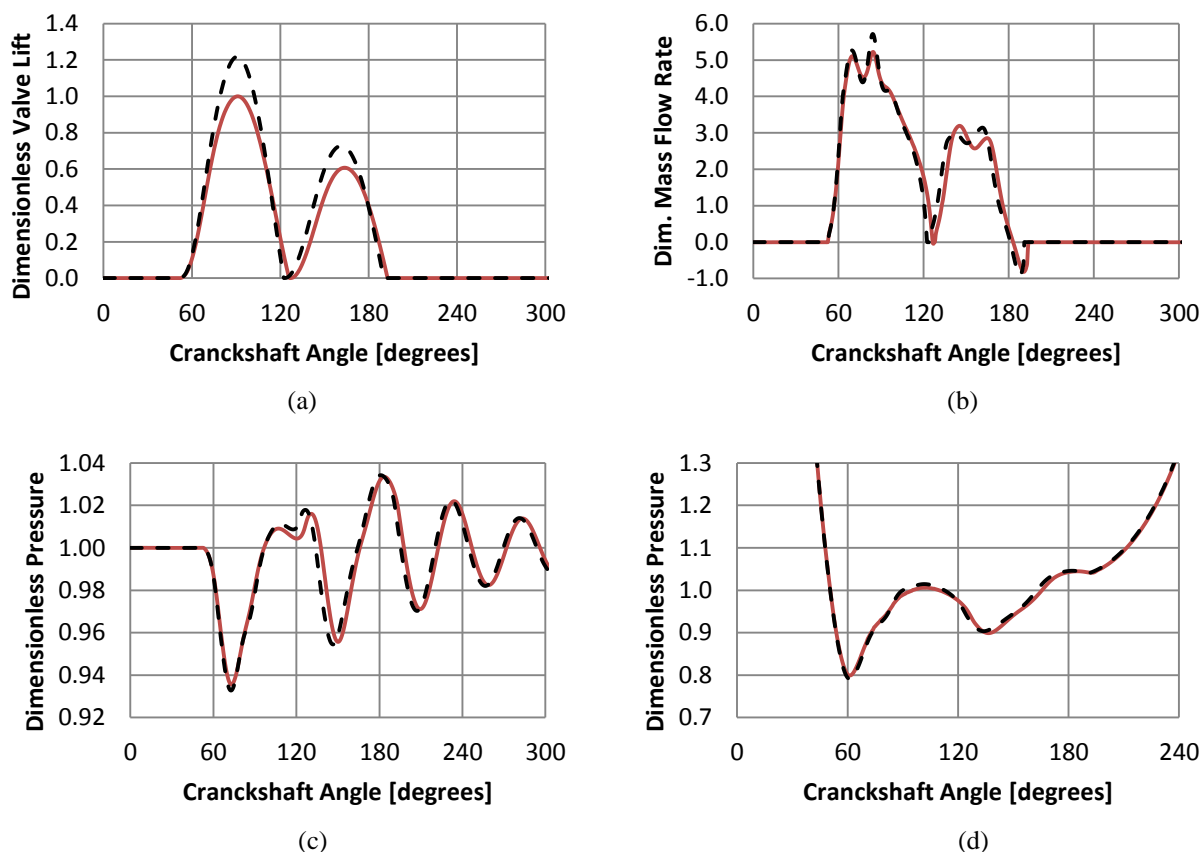


Figure 3: Results for valve displacement (a), mass flow rate (b), pressure inside the cylinder (c) and pressure in the suction chamber (d) during the suction process.

From Figure 3(a), one can observe that the suction valve opens near the crankshaft angle of 50°. As soon as the suction valve opens, refrigerant starts flowing into the cylinder (Figure 3b), causing a pressure drop in the suction chamber (Figure 3c). The valve is seen to bounce towards the seat for the first time around the angle of 90°, with pressure in the suction chamber rising and reaching a maximum just after the angle of 120°. Then a second opening motion gives rise again to a pressure drop in the suction chamber. The valve closes after the bottom dead center (~195° of crankshaft angle) giving rise to backflow in the suction valve, represented by negative mass flow rate in Figure 3b. Pressure pulsations appear in the suction chamber after the valve is closed due to pressure waves travelling forward and backward in the muffler.

Overall, predictions returned by both models are in close agreement. The most significant discrepancy is observed in the valve displacement, with the simplified model predicting larger valve lifts. However, despite such a difference in the valve dynamics, the other results obtained with the simplified are virtually equal to those given by the complete and more expensive model.

Table 1 presents results for the overall performance of the compressor obtained with both models. In general, the deviation between the results of both models was around 1% for the integrated mass flow rate and, consequently, for the cooling capacity and the total suction loss. Considering the results shown in Figure 3 and Table 1, the predictions provided by the simplified CFD model can be considered adequate for the purpose of muffler analysis.

Table 1: Comparative results between the simplified and complete CFD models.

	Simplified	Complete	Deviation
Mass flow rate [kg/h]	2.620	2.593	0.027
Cooling capacity [W]	135.2	133.9	1.3
Total suction loss [W]	1.13	1.11	0.02
Suction valve loss [W]	0.71	0.74	-0.03
Suction muffler loss [W]	0.42	0.37	0.05

4. CONCLUSIONS

A simplified CFD model was developed to simulate the suction process of reciprocating compressors as a compromise between accuracy and computational cost. The following advantages of the simplified model were observed:

- The model returns predictions in close agreement with those obtained with a more comprehensive and computationally expensive model.
- Only a small region of the cylinder is included in the solution domain of the simplified model and this considerably reduces both the effort to prepare the grid and the simulation computational cost.
- The clearance and the geometry of the cylinder can be easily adjusted in the simplified model since the compression process is solved via a lumped model. On the other hand, the complete CFD model requires a new computational grid in the case of any modification in the cylinder geometry.
- The complete model is associated with non-conforming grids at the interface between the suction valve outlet and the cylinder, which can introduce numerical instabilities in the solution procedure. The simplified model does not require such non-conformal grids.

REFERENCES

- Almbauer, R., Lang, W., Nagy, D., Berger, E., 2010, Application of Computational Fluid Dynamics for the Thermodynamic Development of a New Generation of Hermetic Reciprocating Compressor, *Proc. International Compressor Engineering Conference at Purdue*, West Lafayette, USA, C1230, 7 p.
- Bassi, F., Pelagalli, L., Rebay, S., Betto, A., Orefice, M., Pinto, A., 2000, Numerical Simulation of a Reciprocating Compressor for Household Refrigerators, *Proc. Int. Compressor Engrg. Conf. at Purdue*, pp. 97-104.
- Choi, J.K., Joo, J.M., Oh, S.K., Park, S.W., 2000, Smart Suction Muffler Design for a Reciprocating Compressor, *Proc. Int. Compressor Engrg. Conf. at Purdue*, pp. 619-626.
- Elson, J.P. and Soedel, W., 1974, Simulation of the Interaction of Compressor Valves with Acoustic Back Pressures in Long Discharge Lines, *J. Sound and Vibration*, v.34, n.2, pp. 211-220.
- Fagotti, F., Possamai, F.C., 2000, Using Computational Fluid Dynamics as a Compressor Design Tool, *Proc. Int. Compressor Engrg. Conf. at Purdue*, pp. 137-144.
- Lang, W., Almbauer, R., Burgstaller, A., Nagy, D., 2008, Coupling of 0-,1- and 3-d Tool for the Simulation of the Suction Line of a Hermetic Reciprocating Compressor, *Proc. International Compressor Engineering Conference at Purdue*, West Lafayette, USA, C1272, 8 p.
- Matos, F.F.S., Prata, A.T. and Deschamps, C.J., 2002, Numerical simulation of the dynamics of reed type valves, *Proc. International Compressor Engineering Conference at Purdue*, West Lafayette, USA, C15-2.

- McLaren, J.F.T., Tramschek, A.B., Sanjines, A. and Pastrana, O.F., 1975, A Comparison of Numerical Solutions of the Unsteady Flow Equations Applied to Reciprocating Compressor Systems, *J. Mech. Engrg Science*, v.17, n.5, pp. 271-279.
- Pereira, E.L.L., Deschamps, C.J., Ribas Jr, F.A., 2008a, Performance Analysis of Reciprocating Compressors through Computational Fluid Dynamics. *Proc. IMechE, Part E: J. Process Mechanical Engineering*, vol. 222(E4), p. 183-192.
- Pereira, E.L.L., Deschamps, C.J., Ribas Jr, F.A. and Dutra, T., 2008b, Assessment of Simulation Models for Heat Transfer in Suction Mufflers of Reciprocating Compressors, *Proc. 6th International Conference on Compressor and Refrigeration*, 2008, Xi'an, China, 8 p.
- Pérez-Segarra, C.D., Escanes, F. and Oliva, A., 1994, Numerical Study of the Thermal and Fluid-Dynamic Behaviour of Reciprocating Compressors, *Proc. Int. Compressor Engrg. Conf. at Purdue*, pp. 145-150.
- Shiomi, F.H., Takemori, C. K. and Deschamps, C. J., 2009, Comparative Analysis of Different Modeling Approaches for Valve Dynamics of Small Reciprocating Compressors, *Proc. 2009 IIR/IIF International Congress on Compressors*, Papiernicka, Slovakia, 8 p.
- Suh, K., Heo, D.N., Kim, H.S., 2006, CAE/CFD Application for Linear Compressor, *Proc. Int. Compressor Engrg. Conf. at Purdue*, 8 p.

ACKNOWLEDGEMENTS

The authors acknowledge the support provided by EMBRACO and CNPq (Brazilian Research Council) through Grant No. 573581/2008-8 (National Institute of Science and Technology in Refrigeration and Thermophysics).